Test case 3.4 : DNS of flow over 2D periodic hill

Marta de la Llave Plata, Vincent Couaillier

ONERA - The French Aerospace Lab, FR-92322 Châtillon, France

1 Code description

The simulation results presented here have been performed using the discontinuous Galerkin solver *Aghora* [1,2]. The computer code *Aghora* is designed to solve the full set of compressible Navier-Stokes equations in three dimensions. The unstructured solver supports high-order meshes and different types of elements (hexahedra, tetrahedra, and prisms). The DG discretization is based on a modal approach that relies on the use of a hierarchy of orthogonal polynomial functions as basis for the Galerkin projection. The solution in each element is thus expressed in terms of a polynomial expansion, the coefficients of which constitute the degrees of freedom of the problem. In the particular case of parallelepipeds the Legendre basis is used. For tetrahedral elements we use the Dubiner orthogonal polynomial basis, based on Jacobi polynomials. A modified Gram-Schmidt orthonormalization procedure is used for general-shaped elements.

The Lax-Friedrichs and Roe's finite volume schemes can be used to approximate the convective fluxes across the element interfaces. The viscous fluxes can be discretized using the BR2 scheme [3] or the symmetric interior penalty (SIP) method [4]. Time integration can be performed either explicitly or implicitly. Explicit time stepping is based on strong stability preserving Runge-Kutta schemes [5]. More details on the implementation of the implicit approach can be found in [6].

Two parallel strategies have been implemented into *Aghora* which rely on a non-blocking and synchronous point-to-point send method. The first is a classic approach based on the MPI paradigm. The second is a hybrid approach combining MPI and OpenMP. OpenMP is used as a coarse-grain parallelism in which each thread takes care of a subset of elements and faces of the computational grid during the whole iterative time loop. Both strategies have shown the same behaviour in the tests performed on the Curie cluster (PRACE French

Tier-0 system) for a fixed number of cores. Strong scalability analyses have yielded a ratio of 88% between the obtained speedup and the ideal speedup using 21,952 cores and a polynomial degree of 2. Additional savings in the execution time have been obtained by using the hybrid strategy for polynomial degrees $p = 2$ or higher. Typically, for $p = 4$ the gain ranges between 13% and 19%.

The computational results presented in this paper have been obtained by using the pure MPI approach on 896 cores. An explicit third-order Runge-Kutta scheme has been employed for time integration. The discretization of the convective fluxes is based on the Lax-Friedrichs scheme and the SIP method is used for the discretization of the viscous fluxes.

2 Case summary

The simulation has been performed on the Onera production cluster with 896 cores, using a standard MPI strategy relying on non-blocking and synchronous communications. This cluster is an SGI Altix ICE 8200EX platform offering computing nodes composed of 2 six-core Intel Westmere X5675 processors (12 Mo Cache, 3.07 GHz). This cluster incorporates a Lustre parallel distributed file-system and an InfiniBand interconnect network. On this architecture, the TauBench displays a total of $7.341s$ on a devoted node with the parameters proposed in the high-order workshop guidelines. Starting from that score, for the current simulation, the cost per iteration is about 349.75 Work Units.

The time step of the simulation is $\Delta t = 10^{-4}$, normalised by h/u_b. It is worth noting that the severe restriction on the time step arises from the convective time scale of the acoustic waves due to the low value of the Mach number ($M_b = 0.1$), and not from the viscous time scale associated to the penalty term in the SIP method.

The initial solution corresponds to a uniform flow field in the channel. The solution is advanced in time until a statistically steady state is reached. From this point, the flow statistics are gathered over a sufficient number of convective times ($t_c = 9h/u_b$).

3 Meshes

The Gmsh second-order mesh provided on the workshop website and composed of $128 \times 64 \times 64$ elements has been employed. The polynomial degree of the simulation is $p = 3$, which leads to a total number of degrees of freedom of the order of 33.5M.

4 Results

In this section we provide preliminary results for the mean and fluctuating velocity profiles. So far, the statistics have been averaged over only 7 convective times. The simulations are currently being pursued until convergence of the statistical quantities is reached. Figure 1 shows the evolution of the mass flow at the hill crest and the forcing term $dpdx$ over time (in convective time units t_c). The averaged global flow conditions (averaged over the averaging period) normalised by ρ, u_b, and h are :

$$\dot{m}(x = 0) = 9.9997 \cdot 10^{-1},\ Re_b = 2800,\ dpdx = -0.0107$$

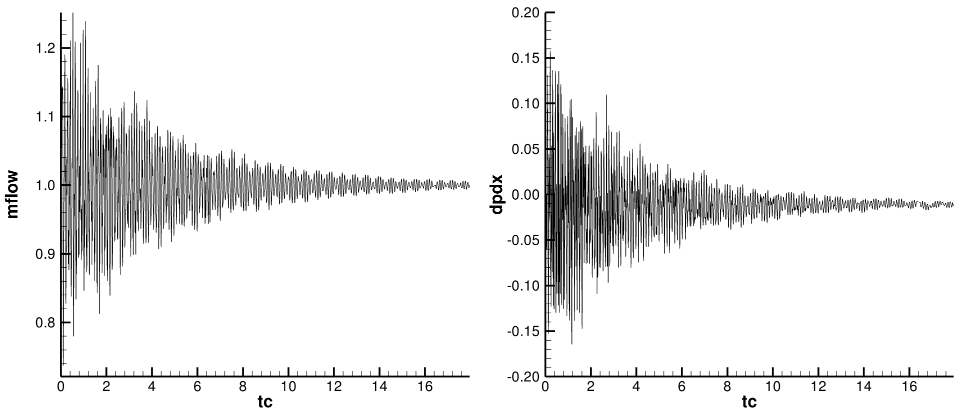

Fig. 1. Evolution of mass flow at hill crest and forcing term with time (normalised by the convective time t_c).

Figures 2 and 3 show the profiles of mean streamwise and vertical velocity at the following spatial locations: $x/h = 0.05, 0.5, 1.0, 2.0, 3.0, 4.0, 5.0, 6.0, 7.0, 8.0$. The results from the $DG - p3$ simulation are compared to the reference DNS results of Breuer et al. [9]. In their paper, the authors used an incompressible second-order finite volume solver (*LESOCC*) on a curvilinear grid composed of 13.1 million points to perform a DNS at $Re = 2,800$.

We can see from Figs. 2 and 3 that, despite the relatively short averaging time used to compute these statistics, a good agreement is found for the mean velocity profiles. The discrepancies found in the fluctuating profiles are due to a insufficient averaging period. As already mentioned above, the averaging process is being pursued until we reach full statistical convergence. The time and spanwise averaged streamwise velocity field is depicted in Fig. 7. The separation length is found to be approximately $x_S/h = 2.2$ and the reattachment length $x_R/h = 5.4$, which is in agreement with the values reported in [9].

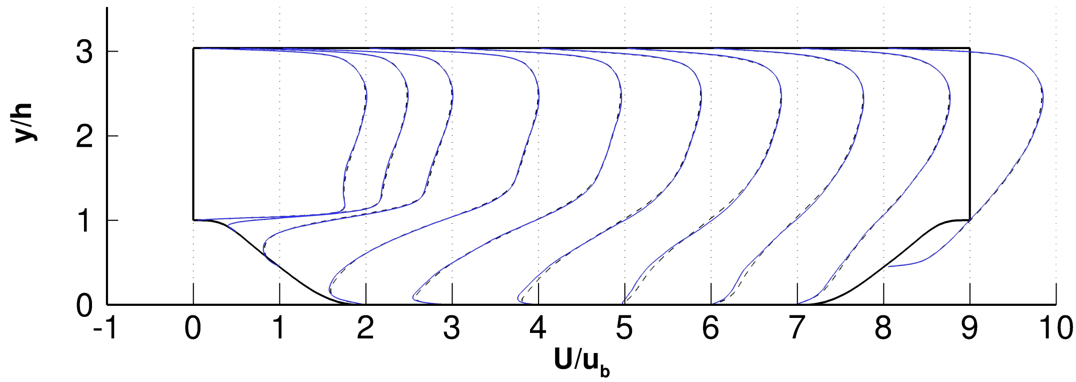

Fig. 2. Profiles of the mean streamwise velocity U/u_b from the DNS computations at $Re = 2,800$. The dashed black lines correspond to the reference results of Breuer et al. The solid blue lines correspond to the results from the $DG-p3$ simulation.

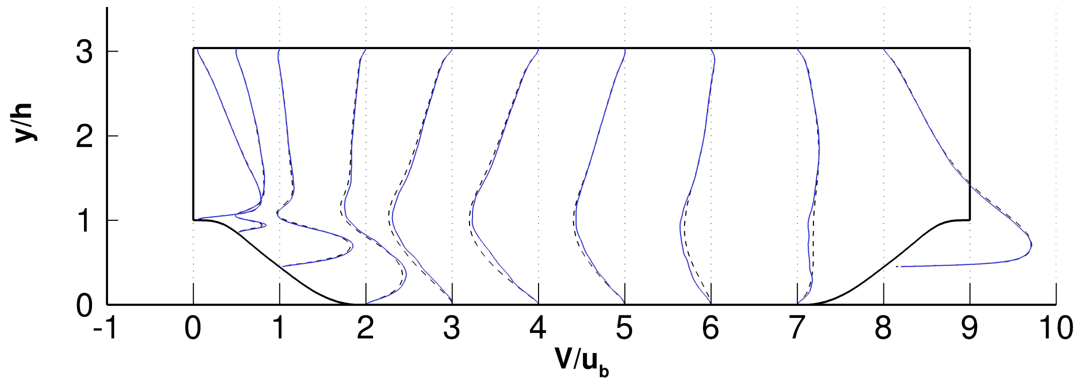

Fig. 3. Profiles of the mean vertical velocity V/u_b from the DNS computations at $Re = 2,800$. The dashed black lines correspond to the reference results of Breuer et al. The solid blue lines correspond to the results from the $DG-p3$ simulation.

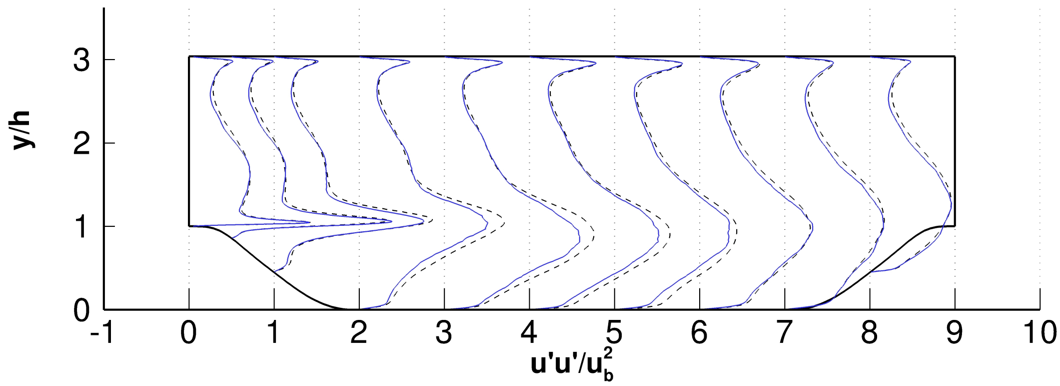

Fig. 4. Comparison of profiles of the Reynolds stresses $u'u'/u_b^2$ from the DNS computations at $Re = 2,800$. The dashed black lines correspond to the reference results of Breuer et al. The solid blue lines correspond to the results from the $DG-p3$ simulation.

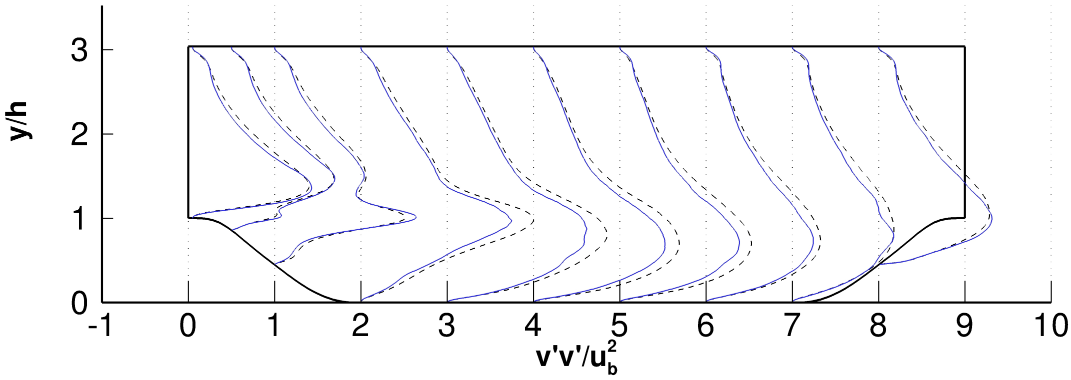

Fig. 5. Comparison of profiles of the Reynolds stresses $v'v'/u_b^2$ from the DNS computations at $Re = 2,800$. The dashed black lines correspond to the reference results of Breuer et al. The solid blue lines correspond to the results from the $DG - p3$ simulation.

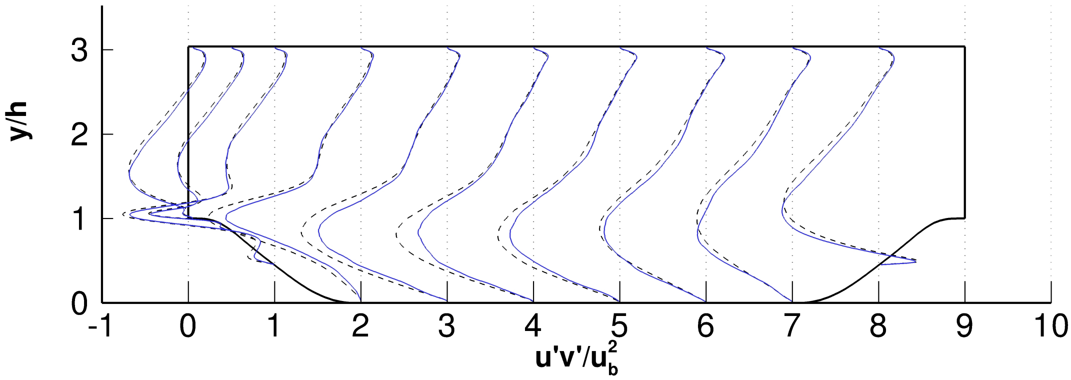

Fig. 6. Comparison of profiles of the shear stresses $u'v'/u_b^2$ from the DNS computations at $Re = 2,800$. The dashed black lines correspond to the reference results of Breuer et al. The solid blue lines correspond to the results from the $DG - p3$ simulation.

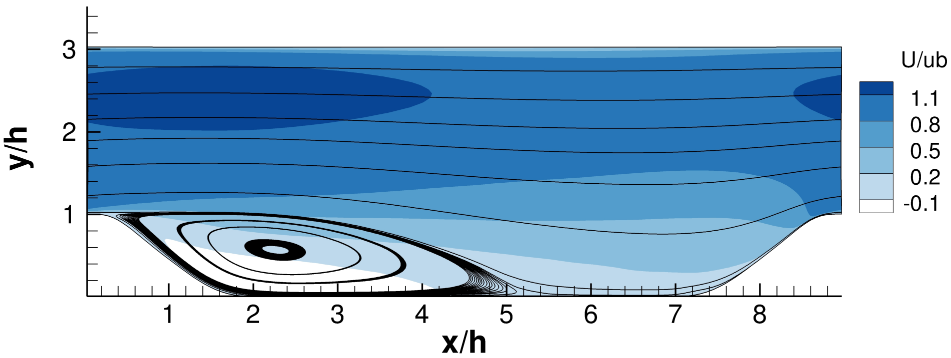

Fig. 7. Streamlines of the time-averaged streamwise velocity field U/u_b from the $DG - p3$ DNS computation at $Re = 2,800$. The colour map corresponds to the time-averaged streamwise velocity U/u_b.

References

[1] F. Renac, C. Marmignon and F. Coquel, Fast time implicit-explicit discontinuous Galerkin method for convection dominated flow problems, Commun. Math. Sci., 10 (2012), pp. 1161–1172.

[2] J.-B. Chapelier, M. de la Llave Plata, F. Renac and E. Lamballais, Evaluation of a high-order discontinuous Galerkin method for the DNS of turbulent flows, Comput. Fluids, 95 (2014), pp. 210–226.

[3] F. Bassi, S. Rebay, G. Mariotti, S. Pedinotti and M. Savini, A High-order accurate discontinuous finite element method for inviscid and viscous turbomachinery Flows, In proceedings of the 2nd European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, R. Decuypere, G. Dibelius (eds.), Antwerpen, Belgium, 1997.

[4] R. Hartmann and P. Houston, An optimal order interior penalty discontinuous Galerkin discretization of the compressible Navier-Stokes equations J. Comput. Phys. 227 (2008), pp. 9670–9685.

[5] R. J. Spiteri and S. J. Ruuth, A new class of optimal high-order strong-stability preserving time discretization methods, SIAM J. Numer. Anal., 40 (2002), pp. 469–491.

[6] F. Renac, M. de la Llave Plata, E. Martin, J.-B. Chapelier, and V. Couaillier, Aghora: A High-Order DG Solver for Turbulent Flow Simulations, IDIHOM: Industrialisation of High-Order Methods - A Top Down Approach, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Springer, to appear in 2014.

[7] H. Bijl, M. H. Carpenter, V. N. Vatsa and C. A. Kennedy, Implicit time integration schemes for the unsteady compressible Navier-Stokes equations: laminar flow, J. Comput. Phys., 179 (2002), pp. 313–329.

[8] M. Baboulin, A. Buttari, J. Dongarra, J. Kurzak, J. Langou, J. Langou, P. Luszczek, S. Tomov, Accelerating scientific computations with mixed precision algorithms, Comput. Phys. Commun., 180 (2009), pp. 2526–2533.

[9] M. Breuer, N. Peller, Ch. Rapp and M. Manhart (2009) Flow over periodic hills - Numerical and experimental study in a wide range of Reynolds numbers, Comput. Fluids, 38(2), pp. 433-457.